for CFA.

SUPERCOMPUTING ASPECTS FOR SIMULATING INCOMPRESSIBLE FLOW

Dochan Kwak

.M.S. T27B, NASA Ames Research Center, Moffett Field, CA 94035,

dkwak@mail.arc.nasa.gov

and

Cetin Kiris

ELORET Inc, ckiris@mail.arc.nasa.gov

The primary objective of this research is to support the design of liquid rocket systems for the Advanced Space Transportation System. Since the space launch systems in the near future are likely to rely on liquid rocket engines, increasing the efficiency and reliability of the engine components is an important task. One of the major problems in the liquid rocket engine is to understand fluid dynamics of fuel and oxidizer flows from the fuel tank to plume. Understanding the flow through the entire turbo-pump geometry through numerical simulation will be of significant value toward design. One of the milestones of this effort is to develop, apply and demonstrate the capability and accuracy of 3D CFD methods as efficient design analysis tools on high performance computer platforms. The development of the Message Passage Interface (MPI) and Multi Level Parallel (MLP) versions of the INS3D code is currently underway. The serial version of INS3D code is a multidimensional incompressible Navier-Stokes solver based on overset grid technology. INS3D-MPI is based on the explicit massage-passing interface across processors and is primarily suited for distributed memory systems. INS3D-MLP is based on multi-level parallel method and is suitable for distributed-shared memory systems. For the entire turbo-pump simulations, moving boundary capability and efficient time-accurate integration methods are built in the flow solver. To handle the geometric complexity and moving boundary problems, an overset grid scheme is incorporated with the solver so that new connectivity data will be obtained at each time step. The Chimera overlapped grid scheme allows subdomains move relative to each other, and provides a great flexibility when the boundary movement creates large displacements.

Two numerical procedures, one based on artificial compressibility method and the other pressure projection method, are outlined for obtaining time-accurate solutions of the incompressible Navier-Stokes equations. The performance of the two methods is compared by obtaining unsteady solutions for the evolution of twin vortices behind a flat plate. Calculated results are compared with experimental and other numerical results. For an unsteady flow, which requires small physical time step, the pressure projection method was found to be computationally efficient since it does not require any subiteration procedure. It was observed that the artificial compressibility method requires a fast convergence scheme at each physical time step in order to satisfy the incompressibility condition. This was obtained by using a GMRES-ILU(0) solver in present computations. When a line-relaxation scheme was used, the time accuracy was degraded and time-accurate computations became very expensive.

The current geometry for the boost pump has various rotating and stationary components, such as inducer, stators, kicker, hydrolic turbine, where the flow is exteremly unsteady. Figure 1 shows the geometry and computed surface pressure of the inducer. The inducer and the hydraulic turbine rotate at different rotational speeds. This causes severe unsteady interactions between rotating and stationary parts. In Figure 2, computational grid for SSME impeller and the performance of MPI/OpenMP hybrid parallel code are presented.

REUSABLE LAUNCH VEHICLE (RLV) TURBOPUMP INDUCER

Rotational Speed : 7850 RPM
Mass Flow : 9093 GPM
Re : 7.99e+7

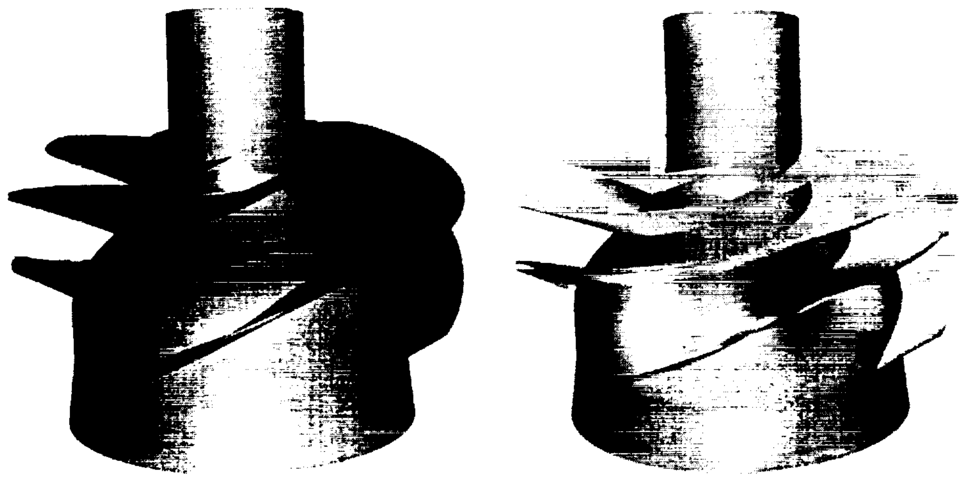

Figure 1: Geometry and surface pressure of boost pump inducer

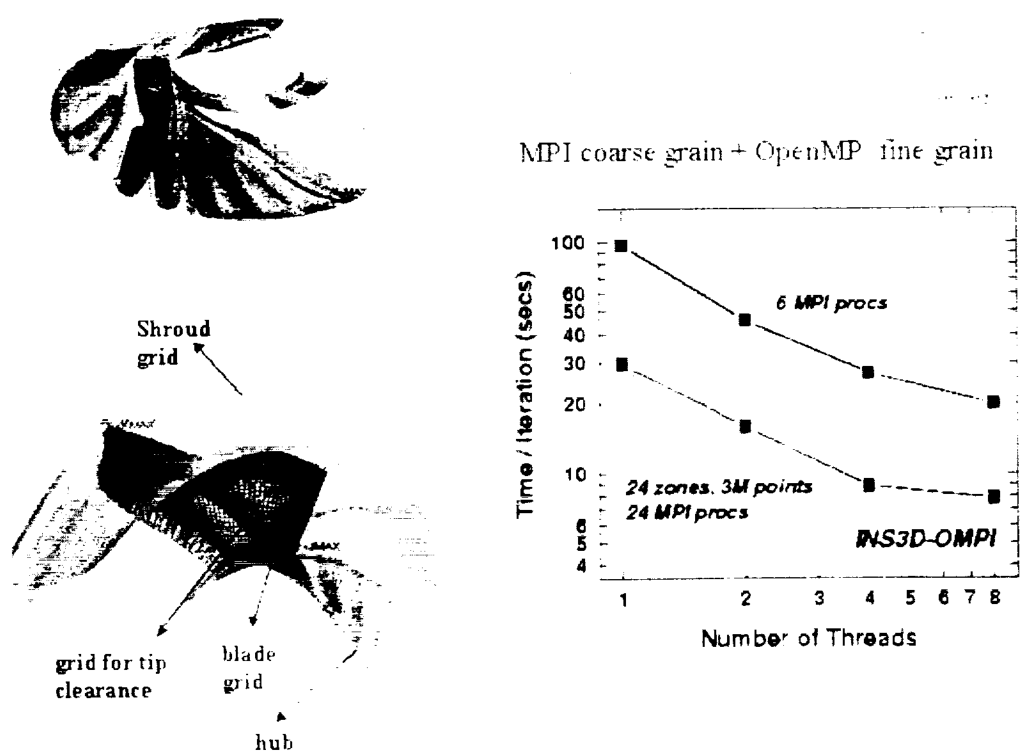

Figure 2. SSME impeller geometry and MPI/OpenMP hybrid parallel code performance.